3-D Solid Modeling and Design

Student Learning Activities
for

Activity #12

PTC

Pro/DESKTOP® 8.0
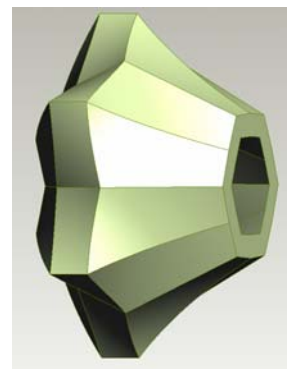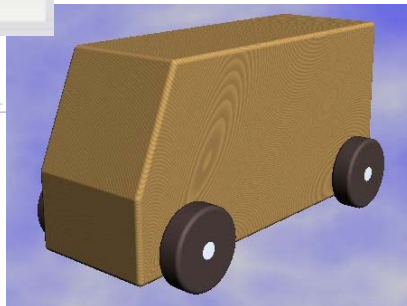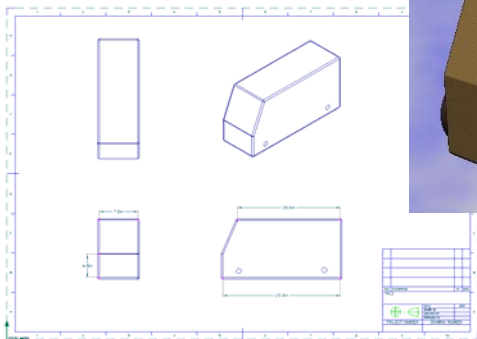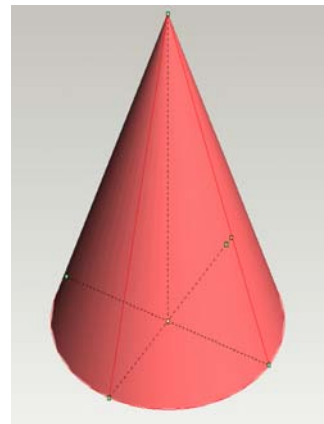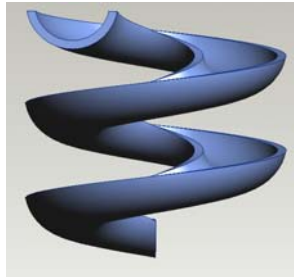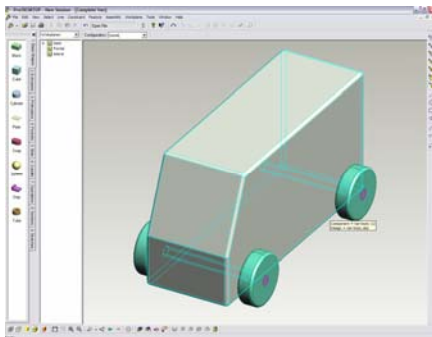
Activity 12:

A Lofty Goal

(Flesch-Kincaid readability level = 6.6)

About the program

Pro/Desktop (called 'PD' from now on) is a powerful software program that allows you to sketch ideas first, and then work on design details later.

This activity will help you:
 ➢ Use the **Loft** command to sculpt different profiles into one solid.

Loft

The **Loft** command merges two or more profiles on different workplanes into a solid. The profiles can have different numbers of edges, and can be on workplanes that are parallel or non-parallel to each other.
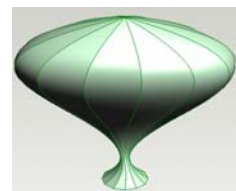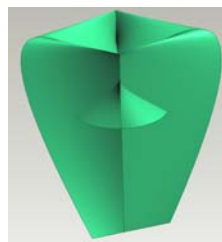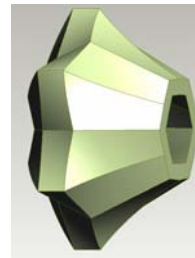
Extrude versus Loft

The main difference between **Extrude** and **Loft** lies in the fact that when you extrude a profile, the profile gets 'taller', but stays in the same shape. With the Loft command, you will be able to make a design that can change shape all up and down the length of the solid. Look below:
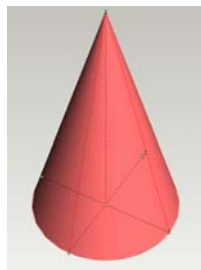
Extrusions   **Lofts**

You need to create two or more profiles, each on a different workplane, to create a lofted solid. You can click and drag a window around all the profiles to make them active (**red**) for lofting. Once you have selected all the profiles to be lofted, PD displays a dashed line connecting each profile to the next. PD also displays a yellow handle on the curve for the profile highlighted in the Profiles list of the dialog box. You can select the grab handle and drag it to change the final shape.

Here we go!

Open a new sketch and rename the **initial** sketch in the base workplane as 'bottom'. Create a new workplane called **top** (in the Workplane pull-down menu) that is parallel and at a 12" offset from the base workplane. Create a new sketch on it called 'top'. Double-click the 'bottom' sketch in the Object Browser Pane to activate that sketch.

In the 'bottom' workplane, sketch an ellipse that is 6" x 3" centered on the Point of Origin.
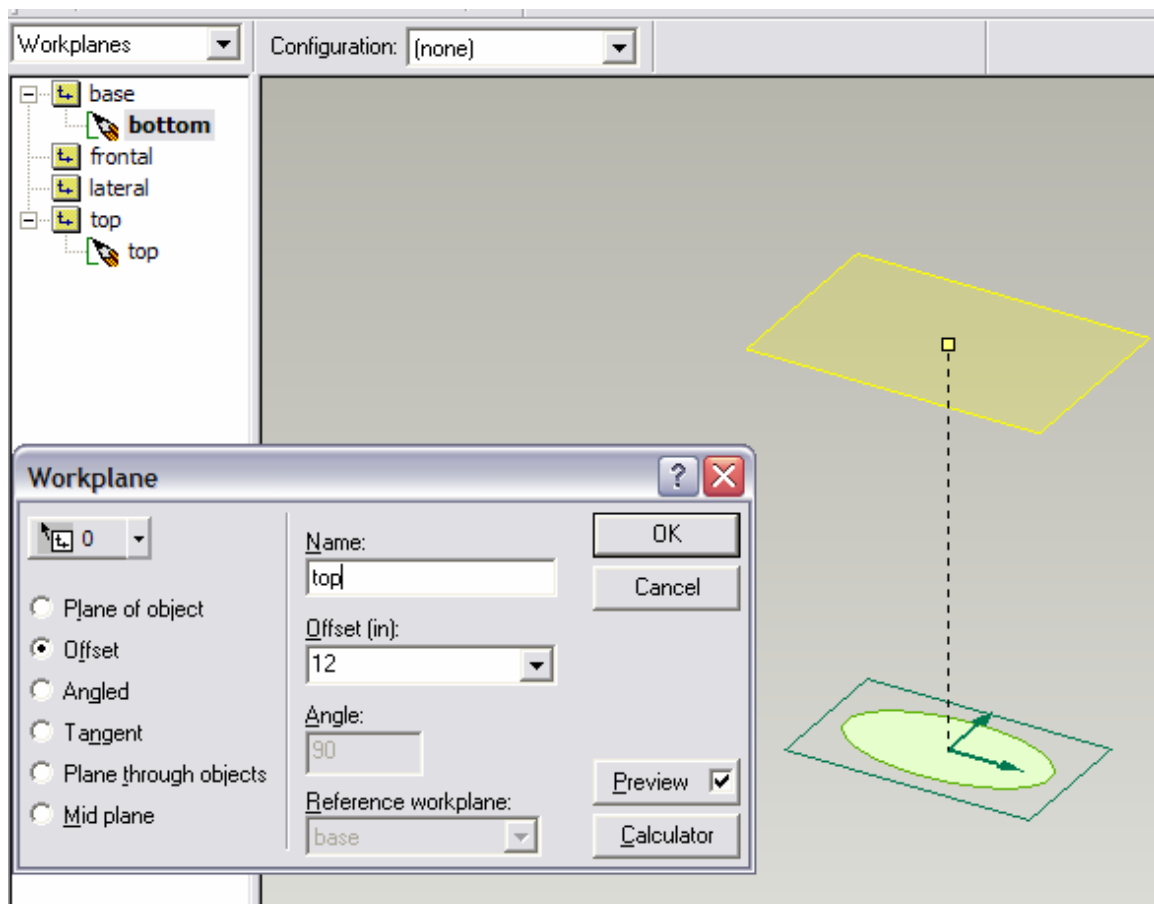
Switch to the 'top' workplane and draw a continuous **Spline** of any shape and size less than 10" across its widest point. Center the Spline as best as you can on the Point of Origin. Make sure the Spline closes so that your profile becomes a valid profile (colored in when finished).
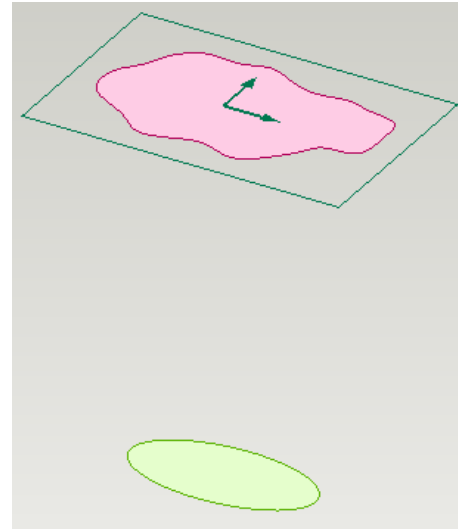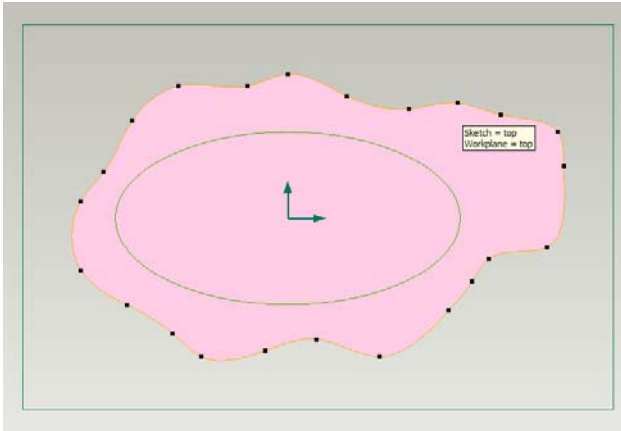
Click and drag your mouse so that you draw a window around both workplanes. When you release the mouse button, **both** sketches become active (lines turn **red**). Select **Loft Through Profiles** in the Features pull-down menu.
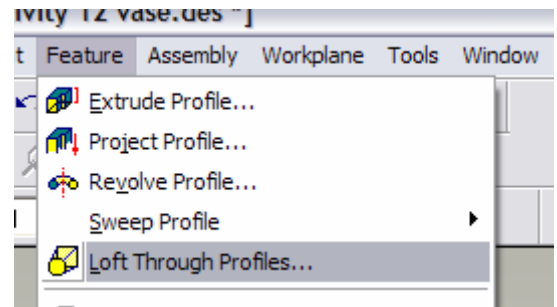
In the dialog box that appears, you will see that, because you dragged a window around the sketches, they became active and are selected (☑). Makes sure the information matches the dialog box below and click OK.
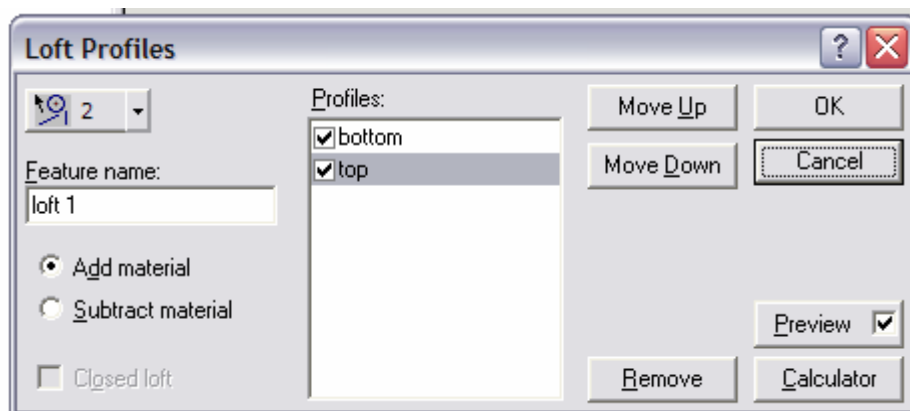
Using the 'Select Faces' command, pick the top face. Click on the 'Shell Solids' icon and either enter .2" in the Offset box, or drag the yellow handle until the readout shows .2" and release.
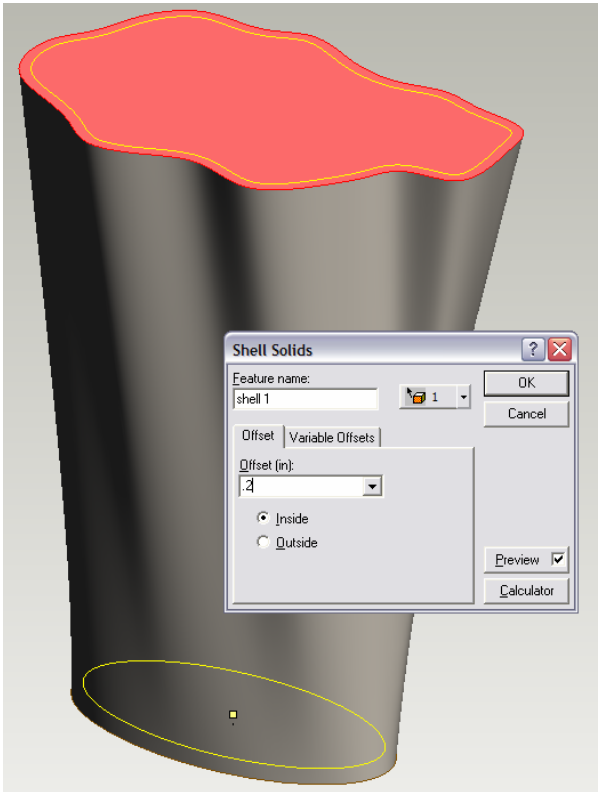
Rotate your flower vase in different directions to get a good look at it.
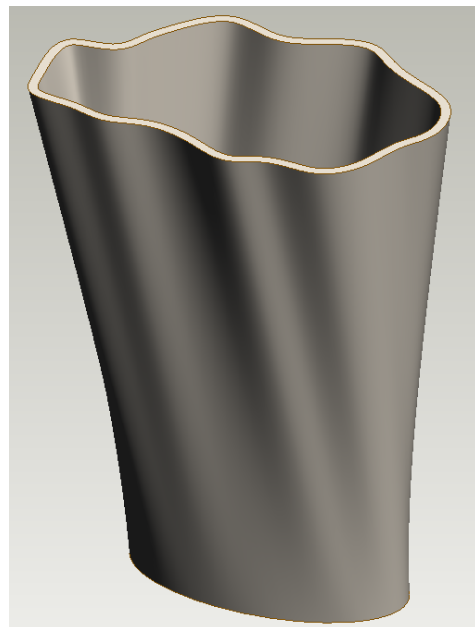
Save your drawing at this point according to the directions from your instructor.

How a square becomes a star!

Begin a new sketch. Rename the 'initial' sketch in the base workplane to 'square'. Create a new workplane at a 60 mm offset from the base workplane and name it 'star'. Create a new sketch in that workplane also called 'star'. In the 'square workplane, create a 30 mm x 30 mm square. Relocate the Point of Origin to the Center of the square (Snap to Grid 15, 15). Switch to the 'star' workplane and Autoscale to the Workplane view. You will still be able to see the square in the 'base' workplane. While it is still visible, relocate the Point of Origin in the 'star' sketch to the center of the square (Snap to Grid 15, 15). While still in the 'star' sketch, draw a 25 mm straight line in any direction starting at the Point of Origin.

Notice that you can still see the square behind the line. Sometimes, it is good to have sketches in other workplanes visible so that you can see that they all line up when you are in the workplane view. For now, we now have the Point of Origin for both sketches as their center and we will not need to see the square anymore. In fact, it will be confusing to stay visible, so we will 'turn it off'.

While in the 'star' sketch, **right-click** on the 'square' **sketch** in the Object Browser Pane. In the dialog box hat appears, click on 'Visible' to un-check it. The square in that workplane will disappear and only the first line in your star will remain.

Now, click on the 25 mm line so that it is active. In the 'Edit' pull-down menu, select 'Duplicate'. In the dialog box that appears, click on the 'circular' tab and enter '5' in the Number box. Leave the Angle at 360. Click OK. Autoscale and select all lines by clicking and dragging a window around all 5. Once all lines are active (**red**), select 'Toggle Construction' under the 'Line' pull-down menu. The lines now become dashed.
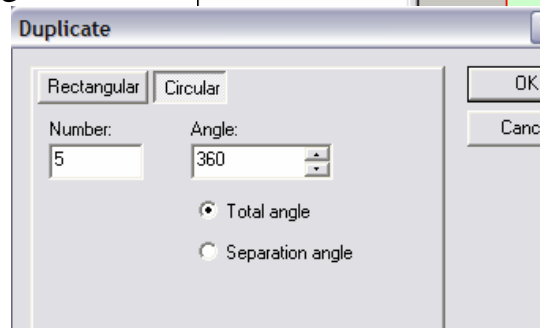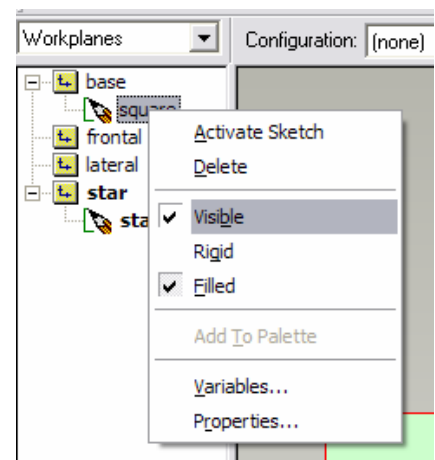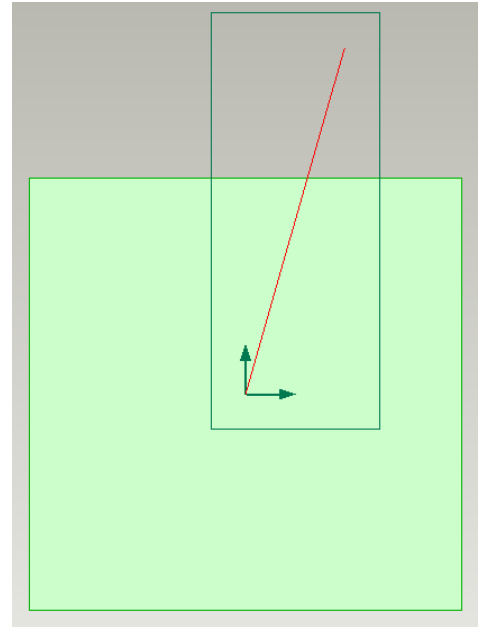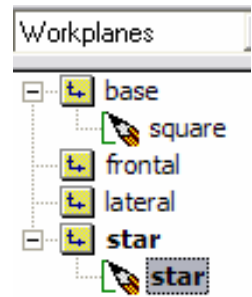
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Save your drawing at this point according to the directions from your instructor.

Using the Line command, connect the ends of all 5 lines to form a pentagon. Once completed, it must be valid (colored in) to continue.

Select all new lines and toggle them into Construction Lines also. Delete the inside construction lines so you are left with a pentagram of Construction Lines. Draw a 5-pointed star inside the pentagon as follows:
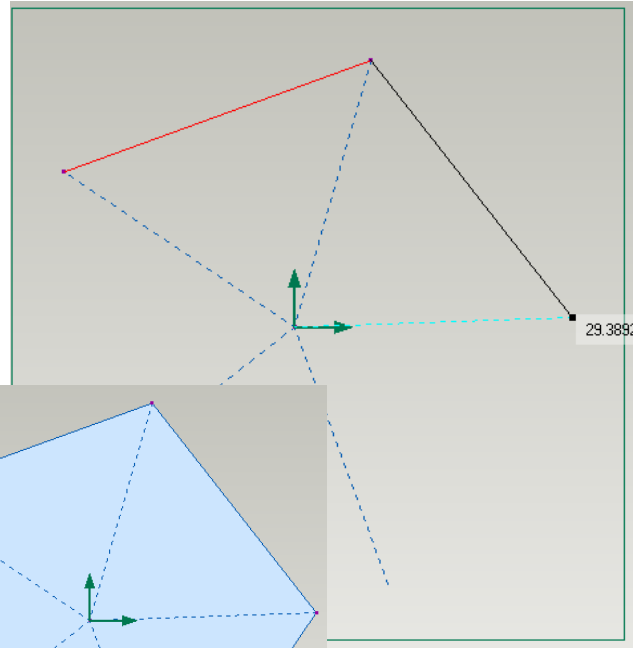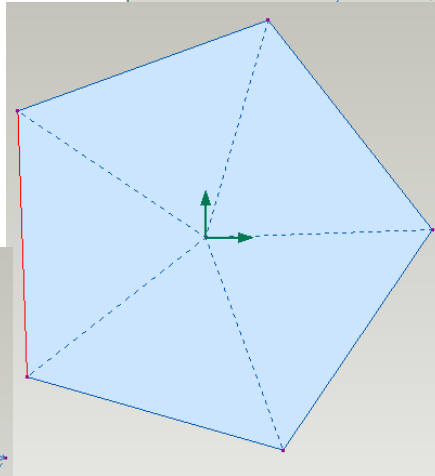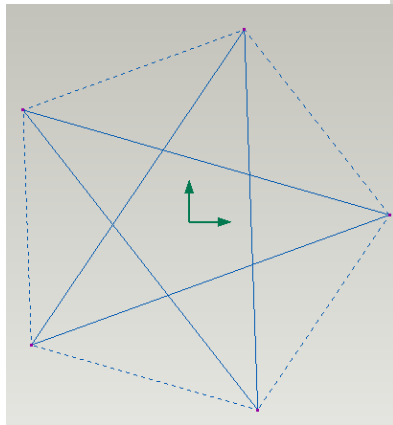
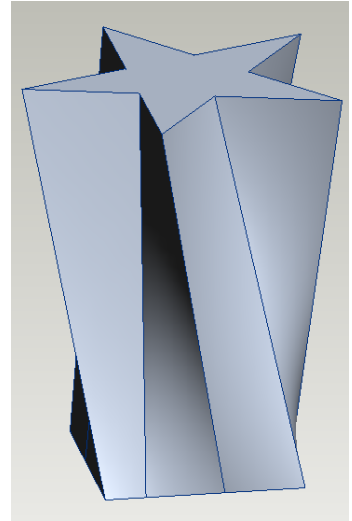**Save** your drawing at this point according to the directions from your instructor.

Using the delete lines icon, remove the center lines of the star to make the star a valid profile.

Right-click on the 'square' sketch and click on the 'Visible' option to make the square appear again. Switch to the trimetric view and half-scale until both the star and the square are visible in the Design Field.
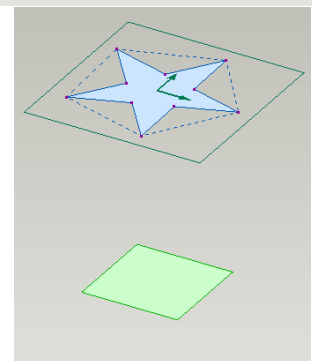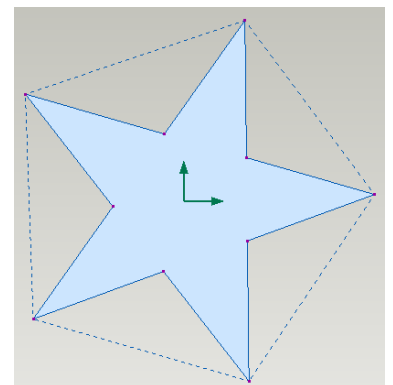
Save your drawing at this point according to the directions from your instructor.
Drag a window around **BOTH** sketches making sure that **all** lines in both sketches are active. Click on the 'Feature' pull-down menu and select 'Loft through Profiles. Make sure that the 'square' and the 'star' profiles are both listed and checked. Click OK. A square has become a star!

Save your drawing at this point according to the directions from your instructor.

You may wish to chamfer/round/ shell solid surfaces to 'dress up your star' before your final save.

Save your drawing at this point according to the directions from your instructor.

One last twist…
        The last loft you will do will have four simple sketches. You will loft from the first through all others to the last one. To do this, create 3 workplanes with their own sketch of the same name as pictured here:
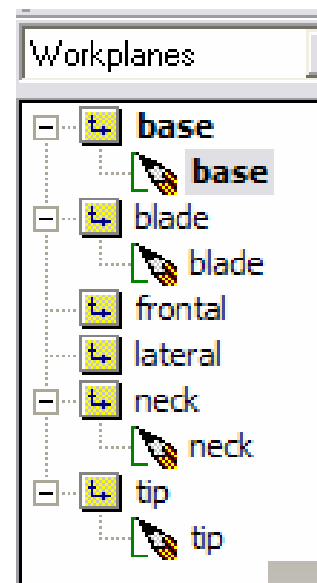
Base workplane
Neck workplane is 6" offset from Base
Blade workplane is 24" offset from Neck
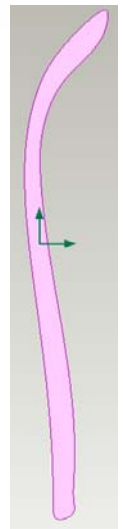Tip workplane is 24" offset from Blade

On the sketches in each of these workplanes, draw the following. All are **centered on the Point of Origin**:

Base: Octagon with 4" long **radial** (starting from the center & pointing outward) construction lines
Neck: 3" Ø circle
Blade: Spline that is approximately .2" wide and 12" long (as close as
                        you can get to looking like this:)➔
Tip: .75" circle

You can turn on and off the Visibility of the sketches as needed. When completed, return to the Base sketch. Switch to the Trimetric view and

half scale until all sketches are visible. Click and drag until all sketches are active. Click on the 'Feature' pull-down menu and select Loft through Profiles **DO NOT CLICK OK AT THIS POINT!!**

Your sketches should look like this: →

You need to know that when you use more than 2 sketches to loft, you will likely want to loft them in a particular order. If you do not specify the order, PD selects it in the order listed in the Loft dialog box. A loft may start at the base sketch, then jump to the top one, then back to the middle. If the order is not selected one at a time in the right sequence, a loft may turn itself 'inside out' by jumping all over from sketch to sketch. Such is the case with this loft shown below. The loft started at the bottom, jumped to the top, then tried to turn inward to the middle sketch. Looks cool, but is often an unwanted mistake:
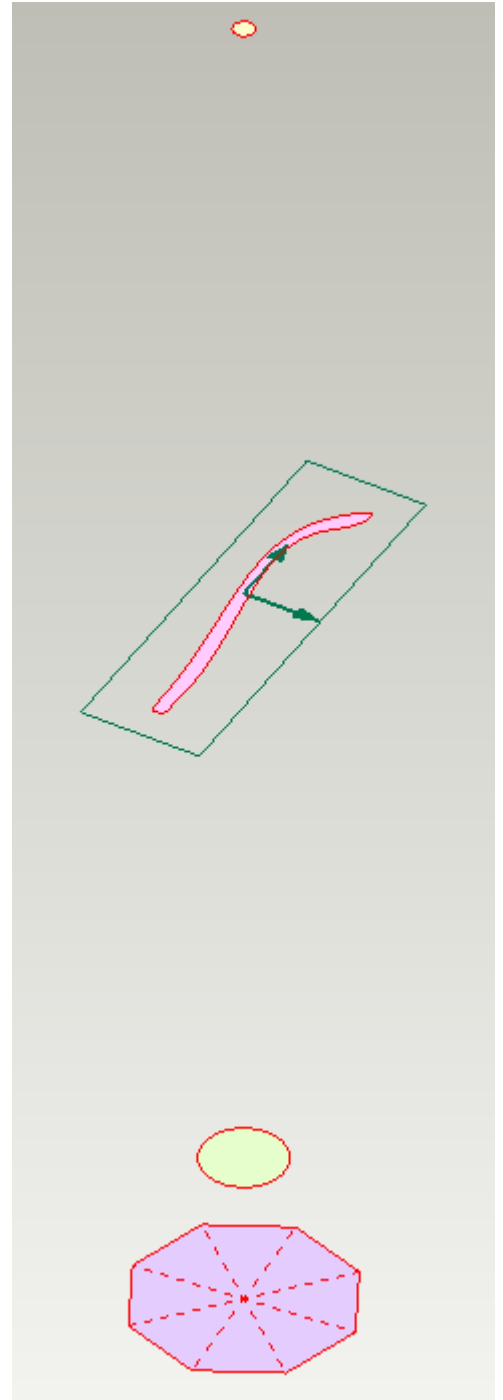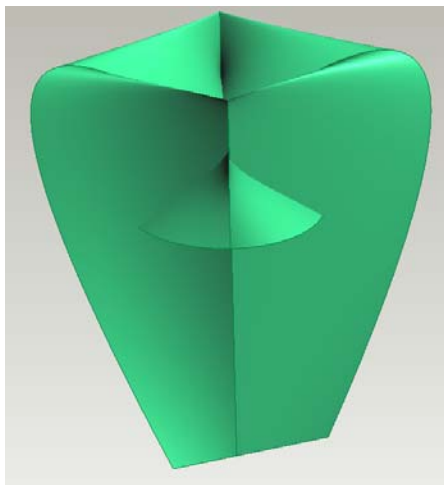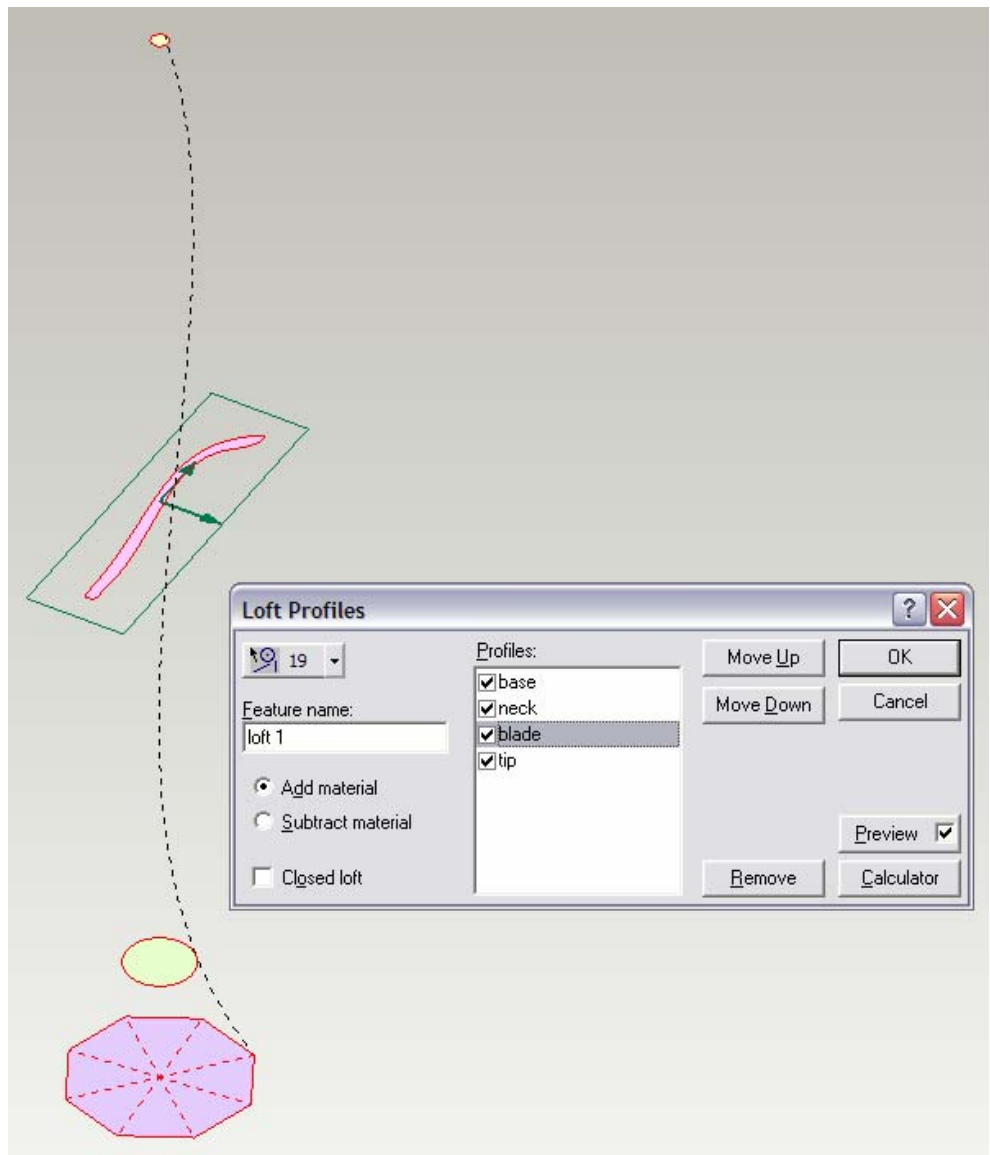
You want to
loft in this order: Base—Neck—Blade—Tip. In the Loft dialog box, notice
the 'Move Up' and 'Move Down' tabs. Select 'Base' and move it up until it
is the first sketch in the list (top). Next, select Neck and move it around until
it is second from the top. Blade is third, and Tip is the bottom of the list.
NOW you can click OK.

Spin your propeller blade around for a
good look.
Save your drawing at this point
according to the directions from your
instructor.